

**ORIGINAL CONTAINS  
COLOR ILLUSTRATIONS**

**N91-10889**

**THE USE OF A NAVIER-STOKES CODE IN THE WING DESIGN PROCESS**

S. Naomi McMillin  
NASA Langley Research Center  
Hampton, Virginia 23665-5225

Abstract of Paper Presented at the  
NASA Computational Fluid Dynamics Conference  
NASA Ames Research Center  
March 6-9, 1989

An ongoing investigation is being conducted in the Supersonic/Hypersonic Aerodynamics Branch at NASA Langley Research Center to determine the feasibility of incorporating the Navier-Stokes computational code, CFL3D, into the supersonic wing design process. The approach taken in this investigation is of two steps.

The first step was to calibrate CFL3D against existing experimental data sets obtained on thin sharp-edged delta wings. The experimental data identified six flow types which are dependent on the similarity parameters of Mach number and angle of attack normal to the leading edge. The calibration showed CFL3D capable of simulating these various separated and attached-flow conditions.

The second step was to use CFL3D to study the initial formation of leading-edge separation over delta wings at supersonic speeds. This study consisted of examining solutions obtained on a  $65^\circ$  delta wing at Mach number of 1.6 with varying cross-sectional shapes. Reynolds number was held constant at 1000000 and the Baldwin-Lomax turbulence model was used. The study showed that through the use of leading-edge radius and/or camber, the onset of leading-edge separation can be delayed to a higher angle of attack than observed on a flat sharp-edged wing.

Based on the geometries studied, three wind-tunnel models are being designed to verify these results. These models are to be tested over a Reynolds number range of  $2 \times 10^6/\text{foot}$  to  $8.5 \times 10^6/\text{foot}$ .

- Fig. 1 The objective and approach for the present investigation.
- Fig. 2 CFL3D code characteristics.
- Fig. 3 Sketches of the wind-tunnel models tested by Miller and Wood in the Langley Unitary Plan Wind Tunnel (NASA TP-2430).
- Fig. 4 The types of flow classified from the Miller and Wood experimental test (NASA TP-2430).
- Fig. 5 The computational test matrix at  $M = 2.8$  superimposed on the chart which defines the flow types as functions of  $\alpha$  and  $M$  normal to the leading edge.
- Fig. 6 A comparison of computational results with experimental data for the  $75^\circ$  delta wing at  $\alpha = 8^\circ$  and  $M = 2.80$  (AIAA 87-2270).
- Fig. 7 The major elements of the incipient separation study (second step of approach).
- Fig. 8 Computational results which quantify the effects of leading-edge radius at  $\alpha = 4^\circ$ .
- Fig. 9 Computational results which quantify the effects of leading-edge radius at  $\alpha = 8^\circ$ .
- Fig. 10 Computational results which quantify the effects of camber at  $\alpha = 8^\circ$ .
- Fig. 11 A summary on the effects of leading-edge radius and camber on a  $65^\circ$  delta wing at  $M = 1.6$ .
- Fig. 12 Major element of proposed wind-tunnel test.
- Fig. 13 Concluding remarks.

## **OBJECTIVE**

- To determine the feasibility of incorporating the Navier - Stokes computational code, CFL3D, into the supersonic wing design process.

## **APPROACH**

- Calibrate CFL3D with experimental data obtained on flat sharp-edged delta wings at supersonic speeds (AIAA 87-2270).

Parameters:

- leading edge sweep
- angle of attack
- Mach number

- Use CFL3D to study the initial formation of leading edge separation on delta wings.

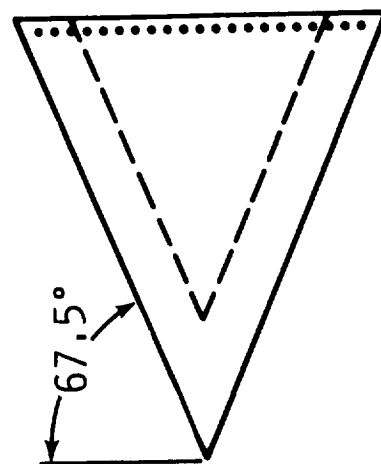
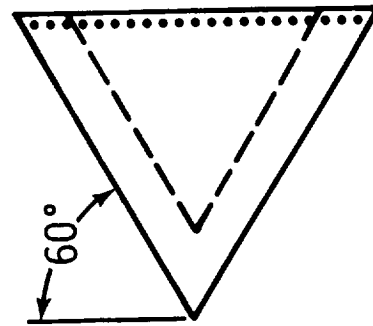
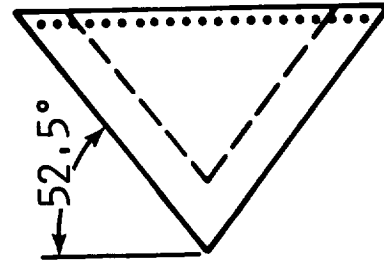
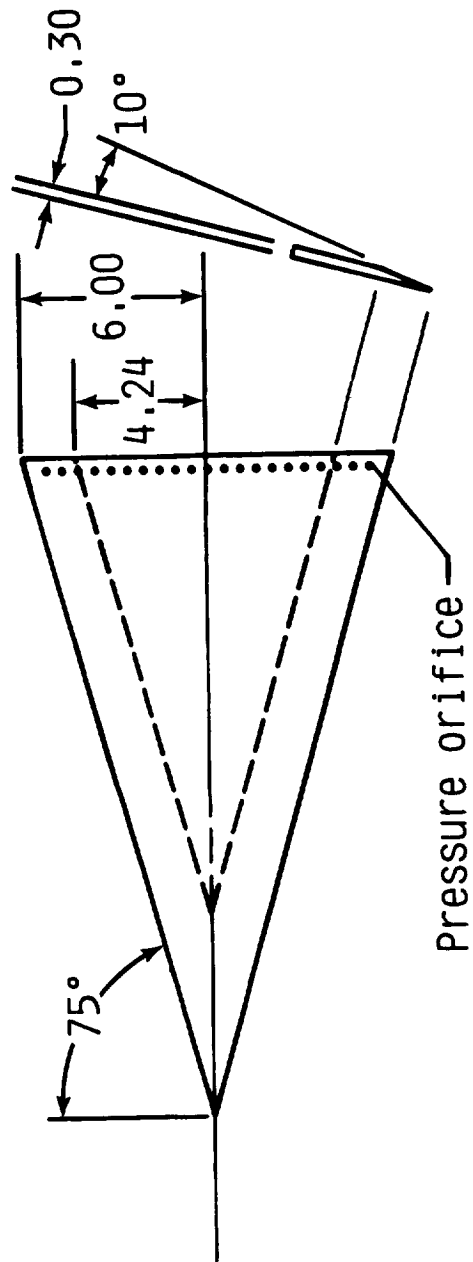
Geometric parameters:

- leading edge radius
- camber

## **CFL3D CODE CHARACTERISTICS**

- Developed at NASA-LaRC by Jim Thomas
- Unsteady conservation-law form of compressible Navier-Stokes equations (thin layer laminar viscous model)
- Laminar viscous model or Baldwin-Lomax eddy viscosity turbulence model
- Upwind-biased spatial differencing implemented in finite volume form
  - Upwind (flux split) for convective terms and pressure
  - Central for shear stress/heat transfer
  - 2nd order accurate
- Implicit time differencing
  - Streamwise relaxation/cross-flow ADI
  - 1st order accurate
- Conical solutions obtained by 3-D solution
  - Single array of conically constructed volumes
  - Inflow equal outflow
- Grids generated using an elliptic grid generation method, 121 x 93 grid required to resolve details of flow patterns

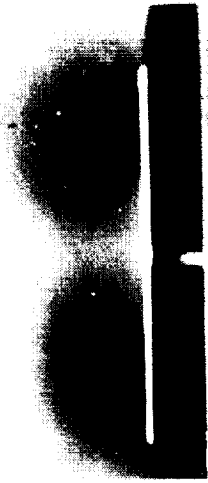
# LEE-SIDE FLOW FLAT DELTA MODELS



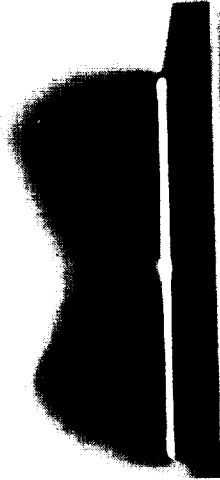
# **FLOW CLASSIFICATIONS**



**SHOCK-INDUCED  
SEPARATION**



**CLASSICAL VORTEX**



**VORTEX WITH SHOCK**



**NO SHOCK/NO SEPARATION**



**SHOCK WITH NO  
SEPARATION**

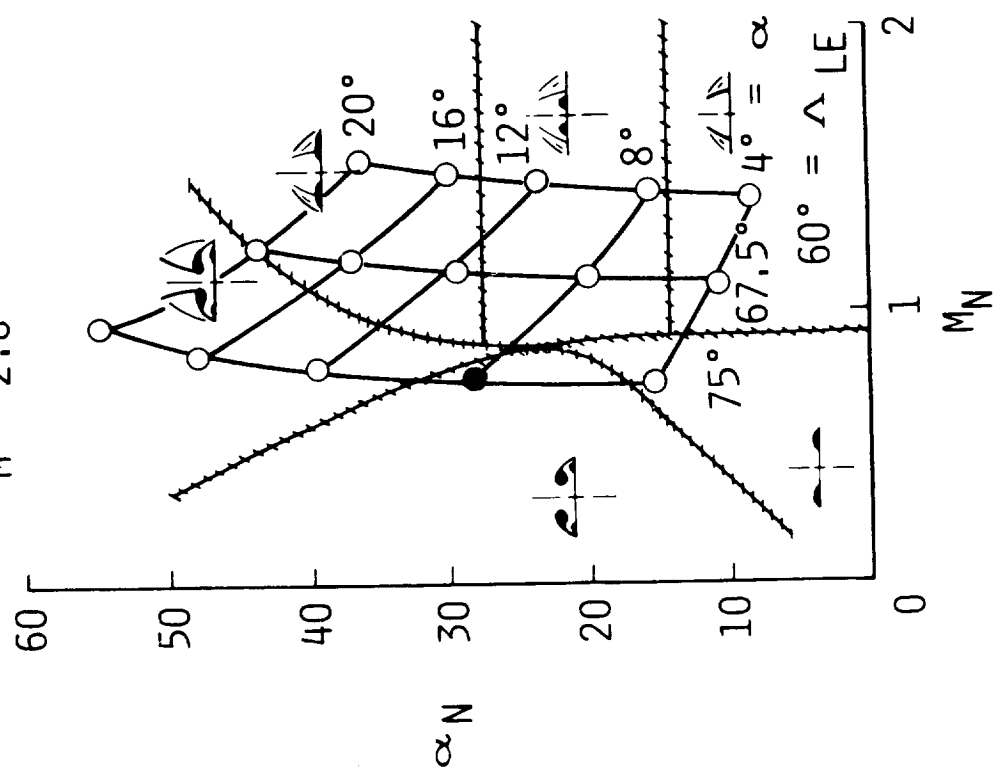


**SEPARATION BUBBLE  
WITH SHOCK**



**SEPARATION BUBBLE  
WITH NO SHOCK**

# THEORETICAL TEST MATRIX







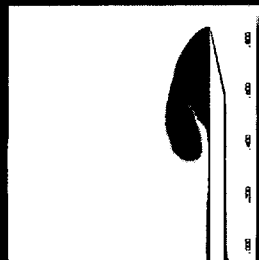
# EFFECT OF VISCOUS MODEL ON NAVIER-STOKES SOLUTIONS

Laminar

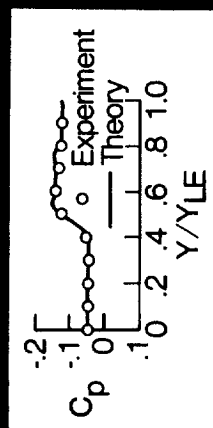
Crossflow Mach number ( $M_\infty/M_\infty$ )

Vapor screen

Total pressure ( $P_T/P_{T\infty}$ )



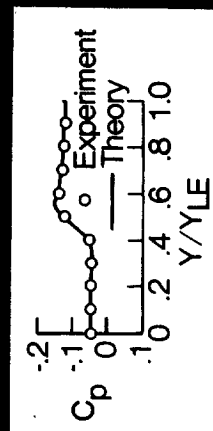
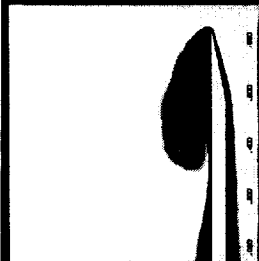
$M = 2.8$   
 $\alpha = 8.0^\circ$   
 $\Lambda = 75^\circ$  Delta wing  
 $R/ft = 2.0 \times 10^6$



Turbulent

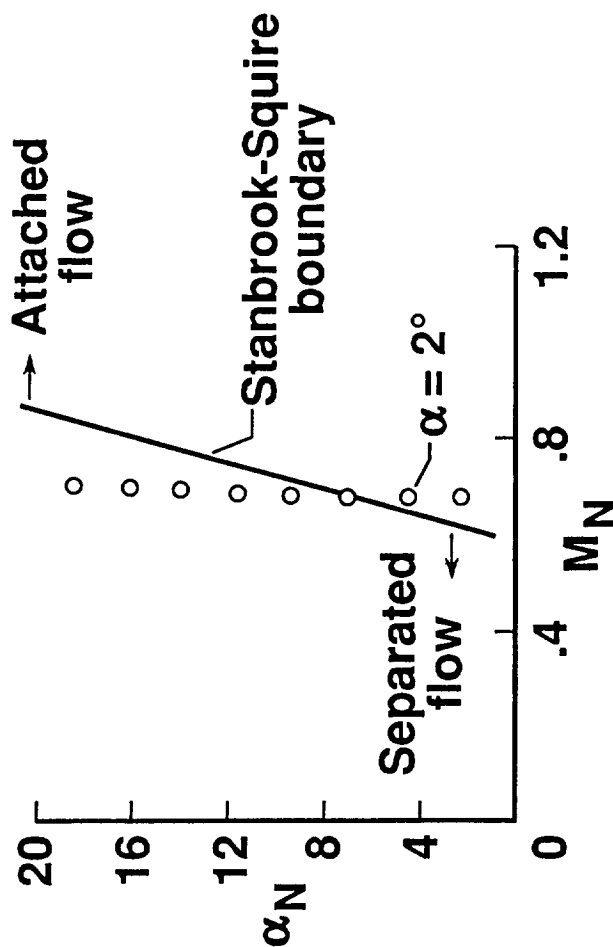
Crossflow Mach number ( $M_\infty/M_\infty$ )

Total pressure ( $P_T/P_{T\infty}$ )





# INCIPIENT SEPARATION STUDY



Examine effects of

- Leading edge radius
- Camber
- Angle of attack
- Reynolds number

On the formation of leading edge separation on a conical  $65^\circ$  delta wing at  $M = 1.6$

Parametric cross-sections

Leading edge radius:

Sharp

$r/(b/2) = 0.0025$   
(20:1 ellipse)

$r/(b/2) = 0.0100$



Camber (20:1 ellipse)  
4° camber

8° camber

10° camber





# EFFECT OF LEADING EDGE RADIUS

$$\alpha = 4^\circ$$

$M = 1.6$   $\Lambda = 65^\circ$   $Re = 1\,000\,000$   
Turbulent boundary layer

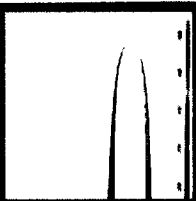
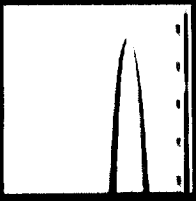
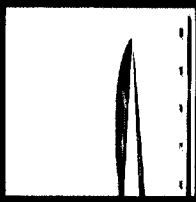
Sharp

$$\frac{r}{b/2} = 0.0025$$

(20:1 ellipse)

$$\frac{r}{b/2} = 0.01$$

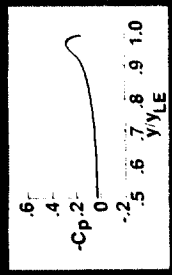
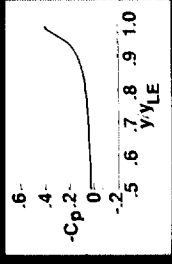
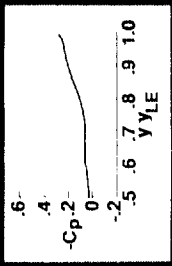
Total pressure:



Crossflow Mach number:



Surface pressure:





# EFFECT OF LEADING EDGE RADIUS

$$\alpha = 8^\circ$$

$M = 1.6$   $\Lambda = 65^\circ$   $Re = 1\,000\,000$   
Turbulent boundary layer

Sharp

$$\frac{r}{b/2} = .0025$$

(20:1 ellipse)

$$\frac{r}{b/2} = .01$$

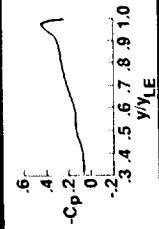
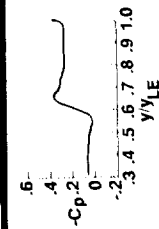
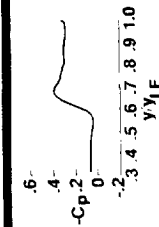
Total pressure:



Crossflow Mach number:



Surface pressure:



ORIGINAL PAGE IS  
OF POOR QUALITY





# EFFECT OF CAMBER

$\alpha = 8^\circ$

$M = 1.6 \quad \Lambda = 65^\circ \quad Re = 1\,000\,000$   
Turbulent boundary layer

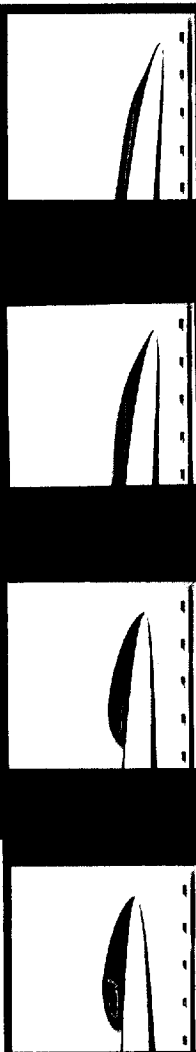
$\alpha_c = 10^\circ$

$\alpha_c = 8^\circ$

$\alpha_c = 4^\circ$

$\alpha_c = 0$

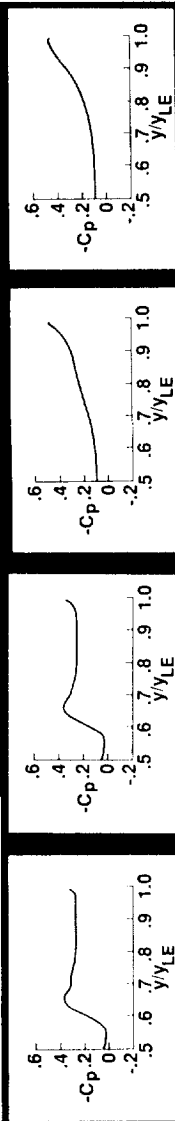
Total pressure:



Crossflow Mach number:



Surface pressure:





# INCIPIENT SEPARATION COMPUTATIONAL STUDY

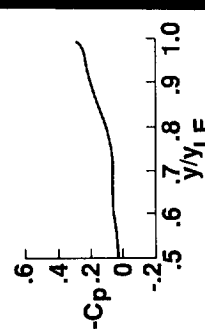
$M = 1.6$ ,  $\Delta = 65^\circ$ ,  $Re = 1 \times 10^6$

Turbulent boundary layer  
color contour plots-crossflow Mach number

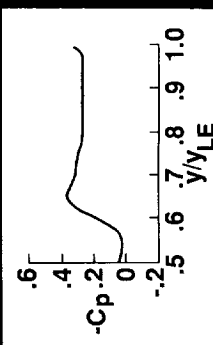
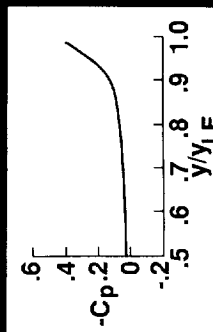
Sharp leading edge

Rounded leading edge  
no camber

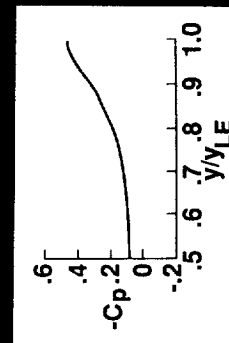
$10^\circ$  camber



$\alpha = 4^\circ$

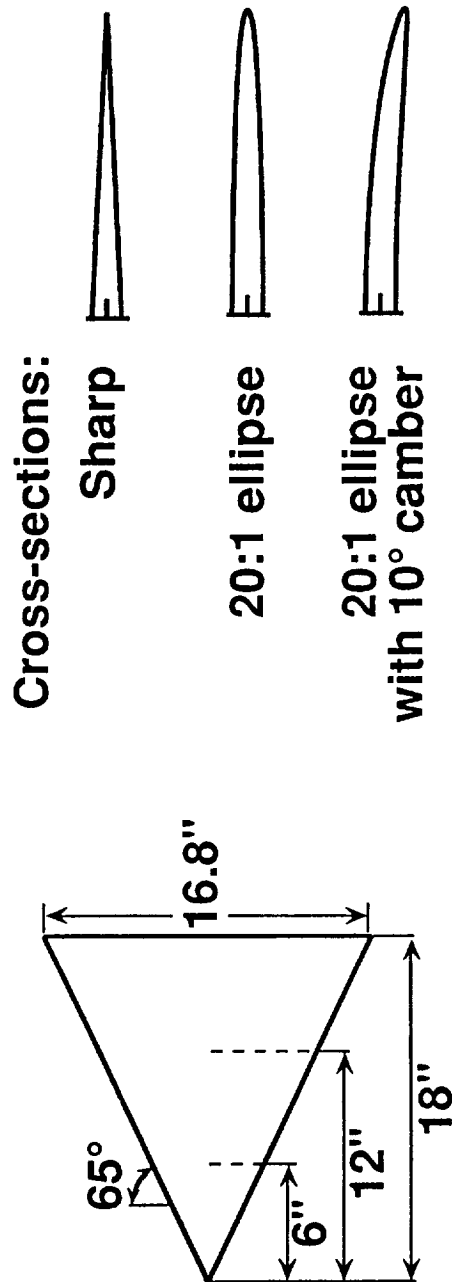


$\alpha = 8^\circ$





# WIND TUNNEL MODELS



## Wind Tunnel Test Description

- $M = 1.6$ ;  $\alpha = 0^\circ$  to  $8^\circ$  (initial conditions)
- Surface pressure data ( $\sim 100$  ports)
- Vapor screen photographs
- Oil flows
- Unitary Plan Wind Tunnel  
 $Re = 2 \times 10^6$  /foot,  $5 \times 10^6$  /foot
- 20" Supersonic Wind Tunnel  
 $Re = 2 \times 10^6$  /foot,  $8.5 \times 10^6$  /foot

## **CLOSING REMARKS**

- **The six distinct flow types observed over delta wings at supersonic speeds have been computationally reproduced using CFL3D (AIAA 87-2270)**
- **CFL3D has been used to quantify the effects of leading edge radius and camber on leading edge separation. Wind tunnel test designed to confirm these results.**